

On the Optimization of SCR System Flue Design

By

**S. M. Cho and J. F. Borowsky
Foster Wheeler Development Corporation
Livingston, New Jersey 07039**

A power plant Selective Catalytic Reduction (SCR) system for flue gas denitrification is normally placed between the economizer and the air heater. A typical flue arrangement in coal plants consists of a horizontal (or inclined) duct where the reagent injection grid and/or fluid mixing device are located, a horizontal-to-vertically-downward turning transition duct, and a vertical catalyst reactor housing section. The flues are designed ideally to obtain uniform flow and reagent concentration distributions at the catalyst inlet surface while minimizing the system pressure drop.

The reagent injection grid consists of many pipes that form several injection zones. The reagent flow rate in each zone can be controlled based on the zonal NO_x level in the corresponding zone. In some designs, an economizer bypass duct from the boiler heat recovery section is provided to condition the flue gas temperature for optimum denitrification reaction and to reduce the potential for ammonium sulfate and bisulfate fouling at the catalyst and downstream equipment during boiler part load operations. A fluid mixing device is also often placed to provide uniform mixing of bypassed flue gas and the reagent gas (such as ammonia-air mixture) with the main flue gas.

The transition duct from the horizontal (or inclined) duct to the vertical catalyst reactor housing section has various configurations depending on the SCR system supplier, available space constraints, and the degree of dust loading. When the flow in the horizontal upstream duct is uniform, the transition duct may ideally be of a triangular shape because this shape maintains constant static pressure as the flow exits vertically downward uniformly. When the upstream horizontal duct flow is not uniform due to upstream flow bends, imperfect mixing, and/or the dust loading is high, then the transition duct may be of a rectangular or trapezoidal configuration with some form of flow guiding device such as turning vanes and flow splitters.

Immediately downstream of the transition duct, but upstream of the catalyst layers, a flow-straightening device called a flow rectifier is normally provided. Because of high dust loading associated with coal firing, the linear velocity of the flue gas stream is normally limited to 20 ft/sec for catalyst erosion consideration. The flue gas velocity vector entering the catalyst layer is straightened with the use of the flow rectifier so as to decrease the potential for catalyst erosion. Generally, the impingement angle of the inlet gas flow for high-dust loading should be on the

order of 10 degrees (or less) with respect to the in-line axis for a practical design. The maximum allowable horizontal velocity component is often specified by the catalyst vendor and is dependent on whether the design is a high dust or low dust SCR application. The rectifier may be a set of vertical plates if the horizontal inlet duct flow is more or less uniform or one-dimensional, and it can be square-shaped plates if the flow is multi-dimensional. The horizontal pitch of the plates should be small in the beginning of the transition duct (because of the abrupt 90-degree turn) and may increase along the depth.

The goal of the SCR system flue design is to provide balanced conditions at the inlet face of the catalyst layer for efficient denitrification reaction. These balanced conditions may be determined by the use of sophisticated Computational Fluid Dynamics (CFD) and physical flow modeling techniques to confirm that the full-scale design will satisfy all of the required flow distribution criteria in the field. Regarding the CFD technique, it can be used to quickly investigate prospective performance by allowing for multiple iterations of analysis for various designs. A two-dimensional model, which requires less computational effort, can be used for each arrangement in order to quickly determine its effects on velocity distribution. A three-dimensional model is generally used once a particular flue/reactor layout is chosen. Each of the designs often has certain desirable features and CFD allows an analysis to determine the relative merits. The CFD code that is often used to model flue arrangements is that of a general-purpose three-dimensional steady-state/transient CFD program. The code solves the Navier-Stokes equations iteratively, using a finite differencing scheme. The code is capable of modeling fluid turbulence, heat transfer, multi-phase (discrete or continuous), mass transfer, and chemical kinetics, although flue analysis often requires only the steady, single-phase flow, turbulence modeling features. Flue configurations can be optimized using a computational fluid dynamics (CFD) code. Some examples are presented.